FreeCAD-CFD Workbench

Tutorial 2: Multi-region mesh generation and porous media using snappyHexMesh

CFD Workbench

WORKBENCH:

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

PREREQUISITES:

WINDOWS:

- Install the binary. All necessary software components are included.
- Install blueCFD build of OpenFOAM (http://bluecfd.github.io/Core/Downloads)

LINUX:

- FreeCAD (*https://www.freecadweb.org/wiki/Install_on_Unix*)
- OpenFOAM (4.1.0 or later) (*https://openfoam.org/download/*)
- Paraview (tested with 5.0.1)
- Gnuplot (tested with 5.0)
- GMSH (2.13+)

For more information, view the CFD workbench README file.

LEAD DEVELOPERS:

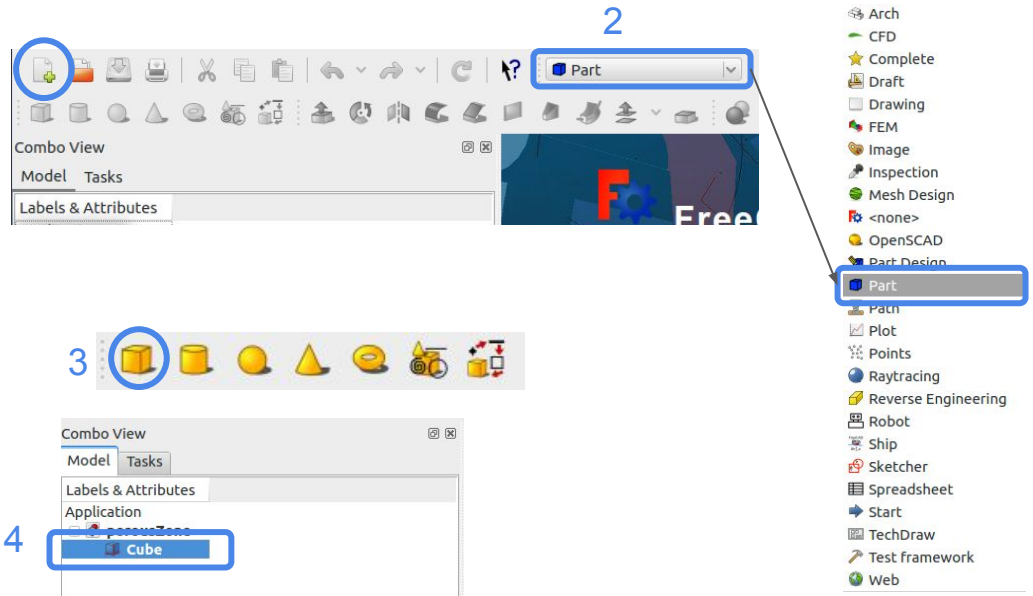
Johan Heyns (CSIR, 2016) jheyns@csir.co.za,
Oliver Oxtoby (CSIR, 2016) ooxtoby@csir.co.za,
Alfred Bogaers (CSIR, 2016) abogaers@csir.co.za,

Part Design

Create Cube

- As part of this tutorial, we are going to create a long thin cube with an internal porous zone and a porous baffle.
- To create the first cube, activate the “Part” workbench.
- Click on the predefined primitive “Cube” icon.
- To change the cube’s length, width and height:
  - Highlight the “Cube” object
  - Within the “Data” tab, change the properties

Length: 500mm, Width: 500mm, Height: 40000mm

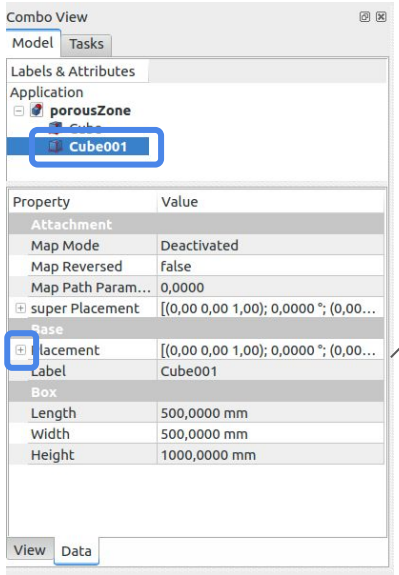


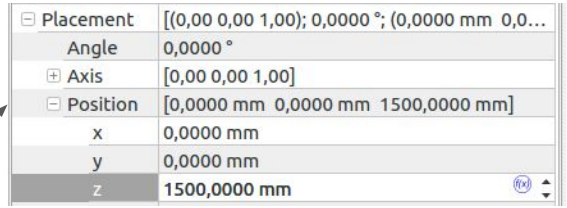
Add 1 More Cube

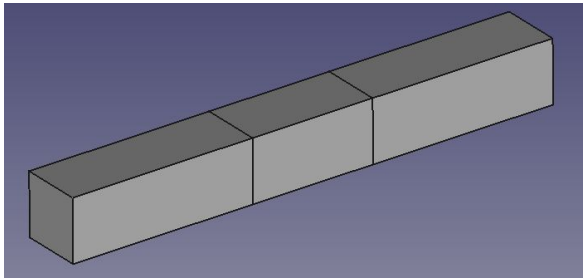
- Create 1 more cube:

Length: 500mm, Width: 500mm, Height: 1000mm

- This cube will become our porous zone and we would like it to be placed in the centre of our large cube.
- To move the cube
  - Click on “Placement” in the data tab.
  - Edit the z-”Position” property to move the cube to z=1500mm



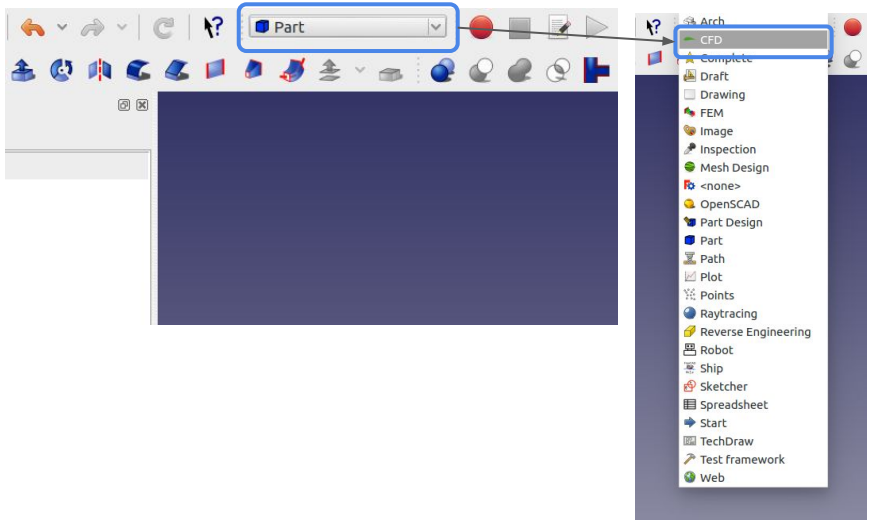




Mesh generation and mesh refinement with snappyHexMesh

Activate Cfd Workbench

- To activate the Cfd Workbench, click on the dropdown menu in the taskbar, and select “CFD”
- Once activated, the Cfd task bar should appear.

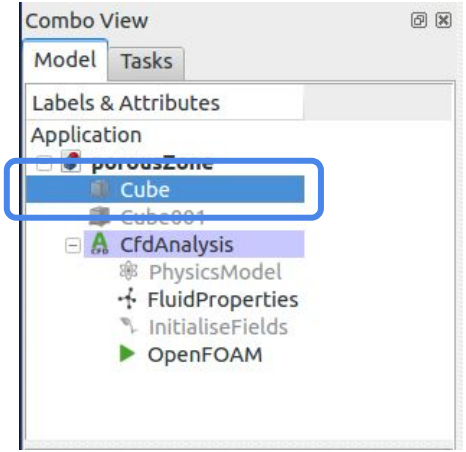


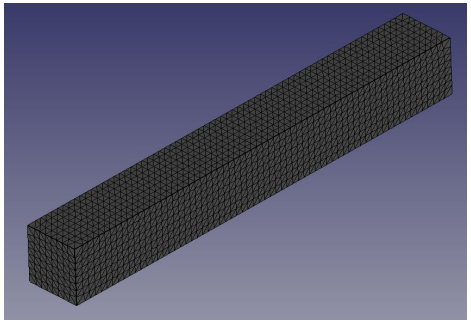
Cfd Task bar:

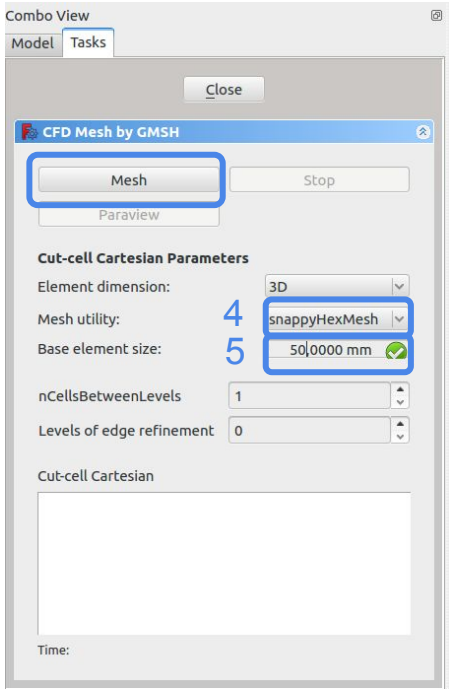
Creating the Initial Mesh

- In order to create a cartesian mesh using snappyHexMesh, we first need to create a new CfdAnalysis
- Highlight the Cube object which will activate the meshing icons.
- Click on the Cartesian Mesh icon.
- Select snappyHexMesh as the mesh utility.
- Set base element size as 50mm (NOTE: if nothing is set a default size will automatically be set based on the object size).
- Click “Mesh”.

NOTE: The tetrahedral mesh is only for illustrative purposes. The real mesh is a cut-cell cartesian mesh. To see the real mesh, please use the paraview post processor (by clicking on the “Paraview” button).

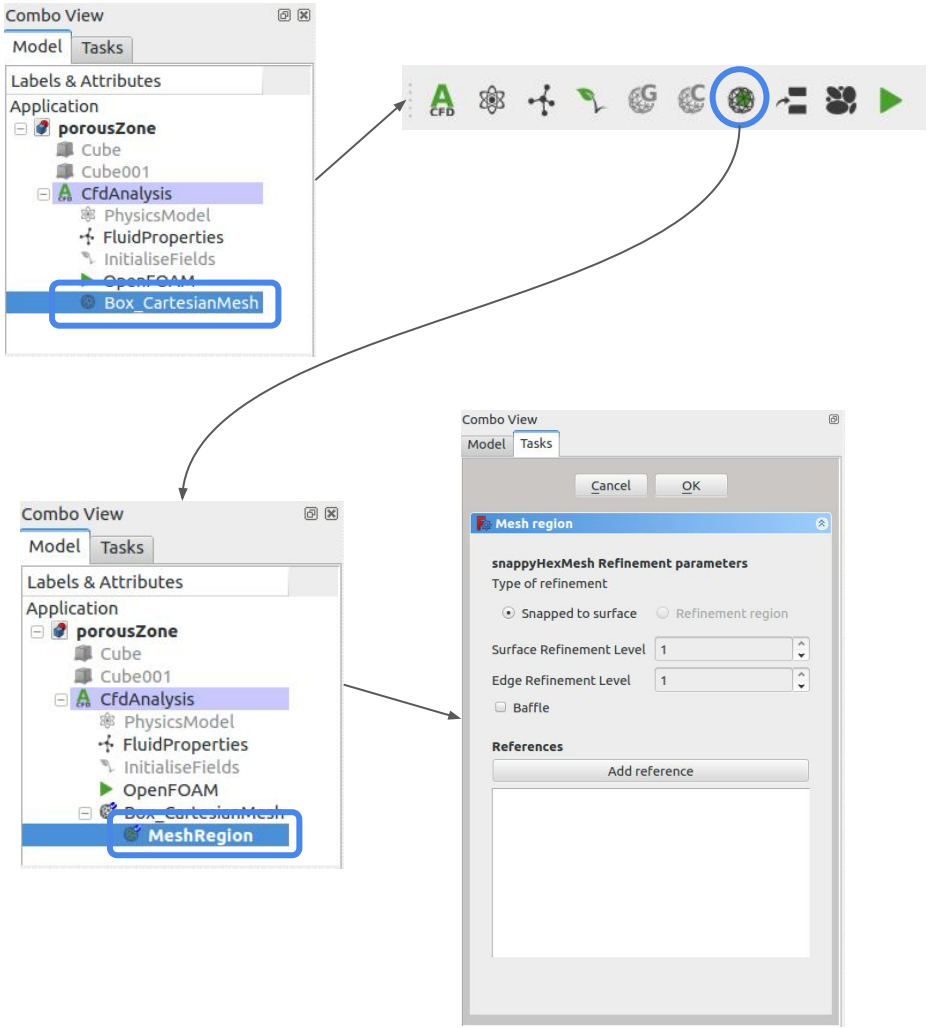






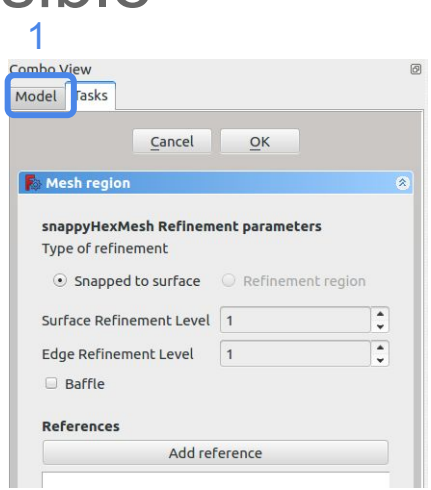
Refining Selected Regions

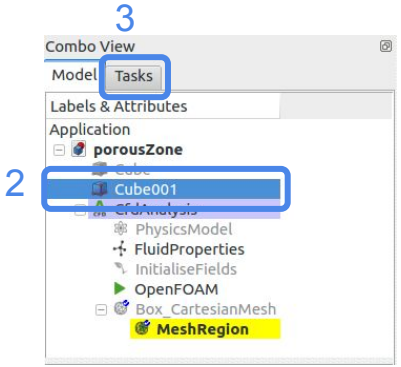
- Highlight the mesh object, which will activate the mesh region icon.
- Click on the mesh region icon (or select “Mesh Region” from the CFD drop down menu).
- A “MeshRegion” object will now have been created (if not visible, expand the “Box_CartesianMesh” object by clicking on the “+”).
- Double click “MeshRegion” which will open the “Mesh region” task panel.
- In this task panel, we can now select shapes in the form faces to be locally refined.

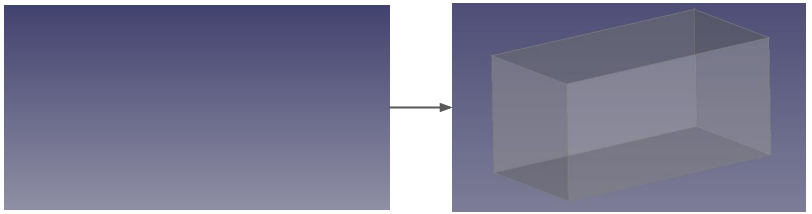


Making specific objects visible

- We wish to now select specific faces to refine, which may not be visible.
- It is possible to only make some of the parts visible, to assist in selecting specific regions, or faces to refine.
- To make specific shapes visible or invisible, click on the “Model” tab.
- Objects can be made visible by clicking on them in the list of objects, and pressing “spacebar”, or right-clicking and selecting “Show Selection” or “Hide Selection”.
- Make the middle cube, the *Cube001* object visible.
- With the middle cube now visible, return to the mesh region task panel, by clicking on the “Tasks” tab.



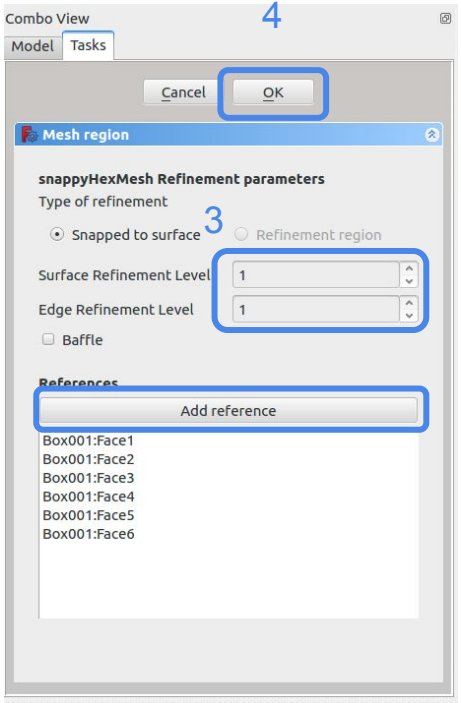


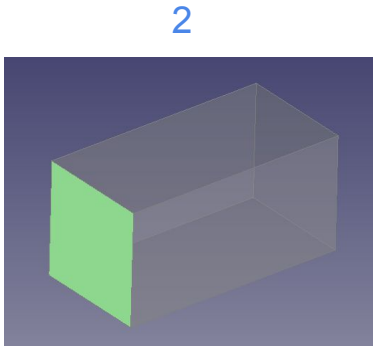


Select regions and setting for refinement

- We wish to refine all the faces within the central region.
- Click “Add reference”, then systematically go about selecting all of the faces belonging to the *Cube001* object.
- Set refinement levels to 1.
- Click Ok.

NOTE: Refinement level when using snappyHexMesh relates to the number of times the mesh falling along the chosen surfaces are to be refined. A value of 1, means the mesh along the chosen surfaces will be refined once, and therefore will be half the size of the original mesh.

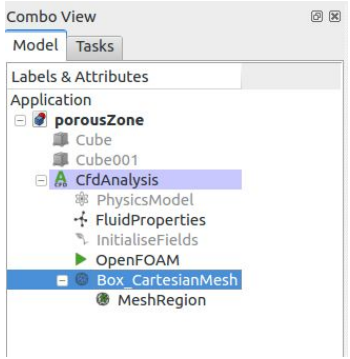


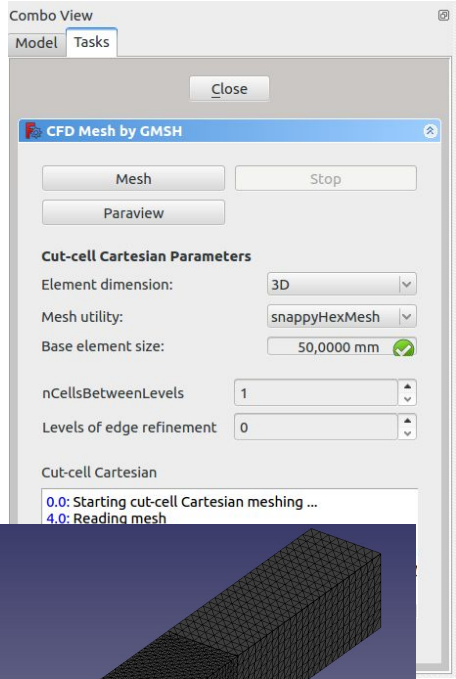


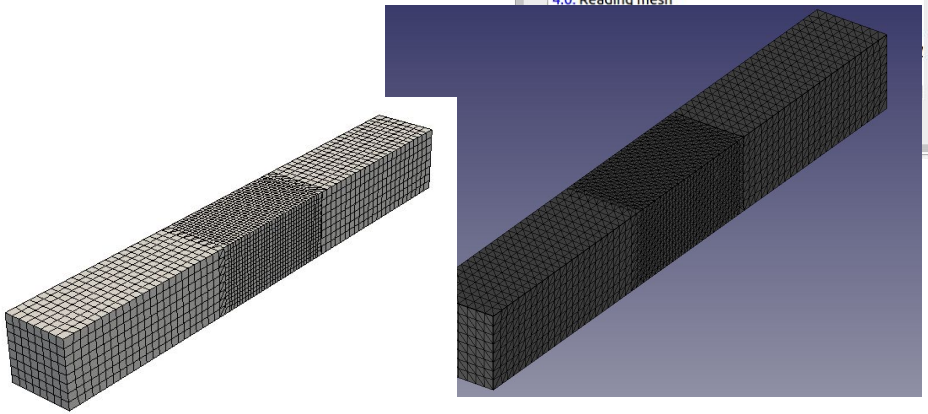
Re-compute the new mesh

- In the list of objects, double-click on the parent (“Box_CartesianMesh”) mesh object (or right-click and select “Transform”).
- Click “Mesh” to re-compute the mesh with the new refined region.
- The new mesh should now be locally refined along the centre.

NOTE: The tetrahedral mesh is for representative purposes only. To view the true cut-cell cartesian mesh click on the “Paraview” button.





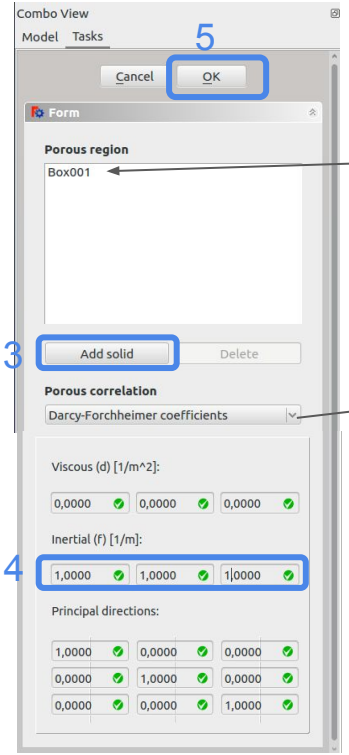


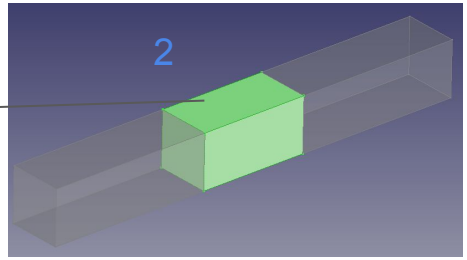
Porous-zones

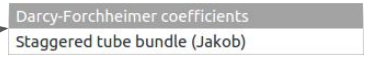
Adding porous-zone

- Click on the porous-zone icon.
- For the test problem, we wish to make the central cube a porous region.
- Select the central cube and click “Add solid”.
- **NOTE:** Porosity will be added to all the cells contained inside, and intersected by the selected shape (in our case “Box001”).
- There are two types of porosity approximations currently included
    - Darcy-Forcheimer
    - Jakob staggered tube bundle

Tip: porous zones are coloured purple once selected.





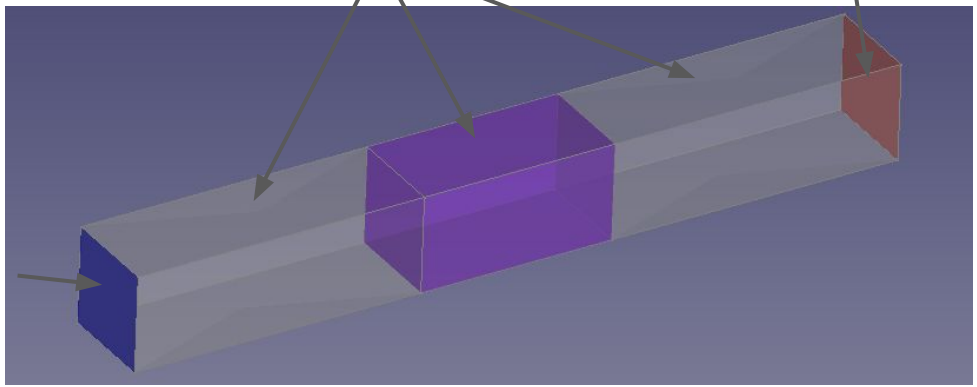


Adding the remaining boundaries

- Following the instructions provided in Tutorial #1, add boundary conditions, for the inlet, side walls and outlet.
- The remaining settings include:
  - Fluid: air
  - Initialise with potential flow

Walls:
Slip

Outlet:
Static pressure
P = 0

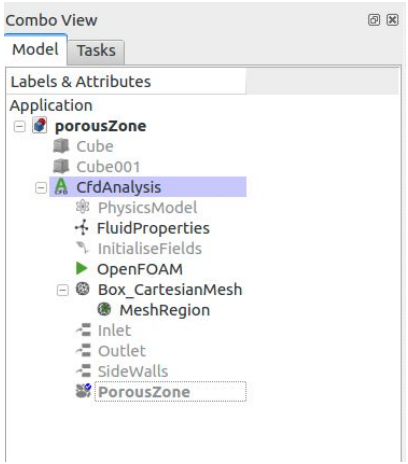


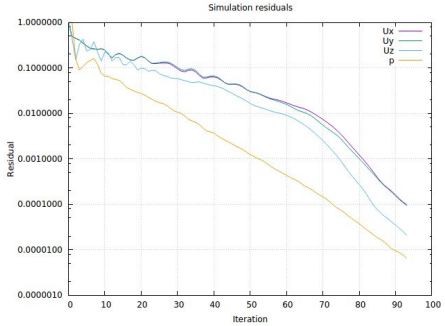
Inlet:
Uniform Velocity
U = (0,0,1 m/s)

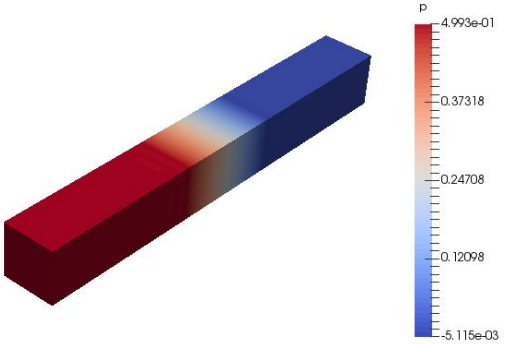
Porous zone results

- With the problem setup we can now write the test case, and run the solver.
- We show here the residual plot and pressure results for the porous zone test problem.
- Notice the large pressure drop across the porous zone.

NOTE: The pressure solved for by OpenFOAM is p = P/density.



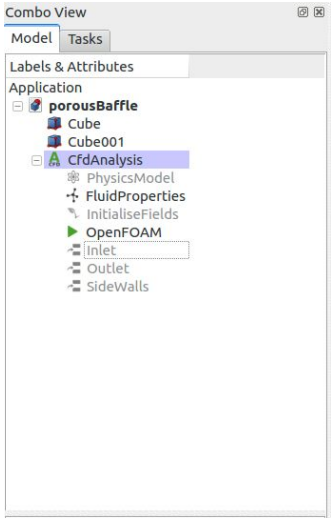


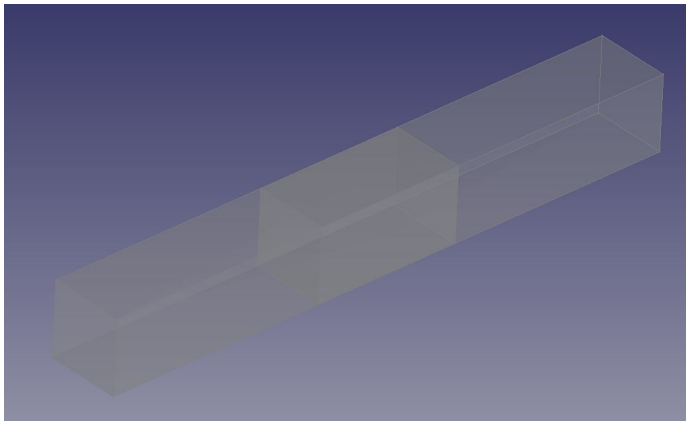


Porous baffle

Creating a porous baffle

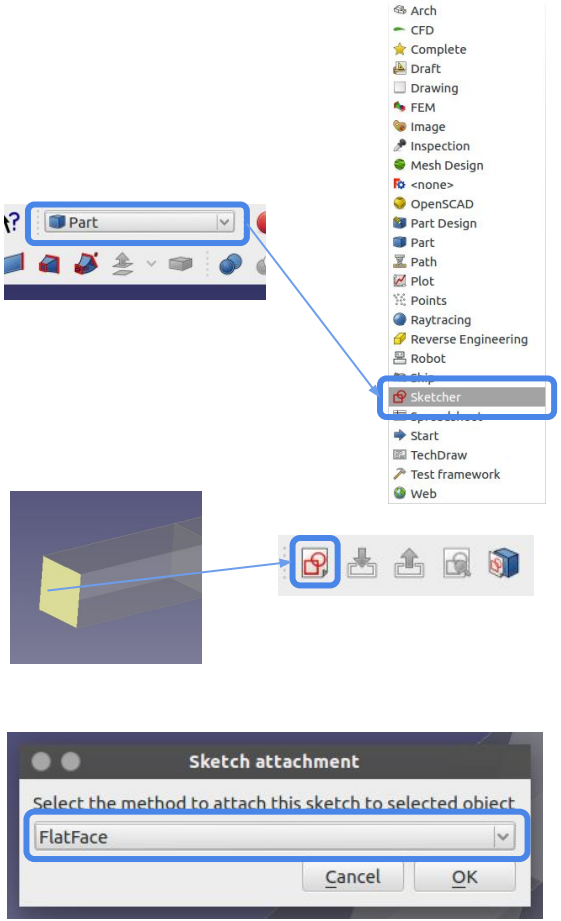
- A porous baffle is created by setting the “Baffle” option within snappyHexMesh refinement regions, and then adding a corresponding “baffle” boundary condition.
- We reset our previous simulation and remove the previously created porous zone.
- We now need to create a face, which we will position within the centre of our analysis domain.





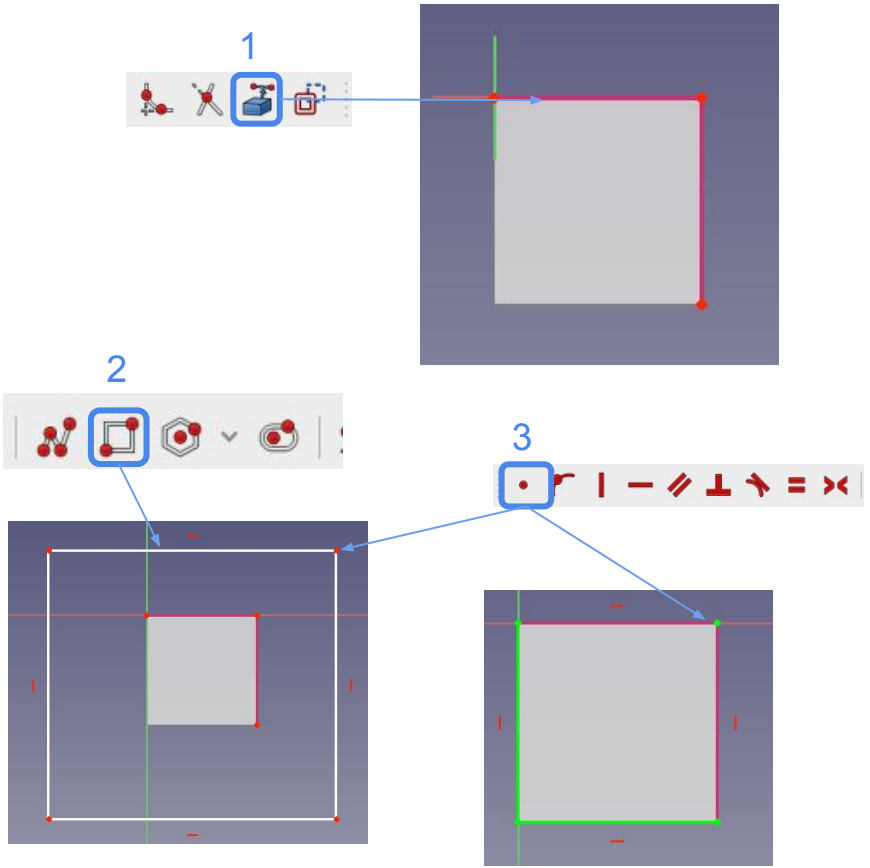
Initialise the sketch

- There are multiple ways of creating faces.
- We are going to do so by creating a sketch of a rectangle, and creating a face from the edges.
- Activate the sketcher workbench.
- Highlight the leftmost face, and click on the “sketcher” icon. This will open a prompt asking where to place the sketch.
- Select “FlatFace”, this is the current face you have highlighted, allowing us to create a sketch directly onto the highlighted face.



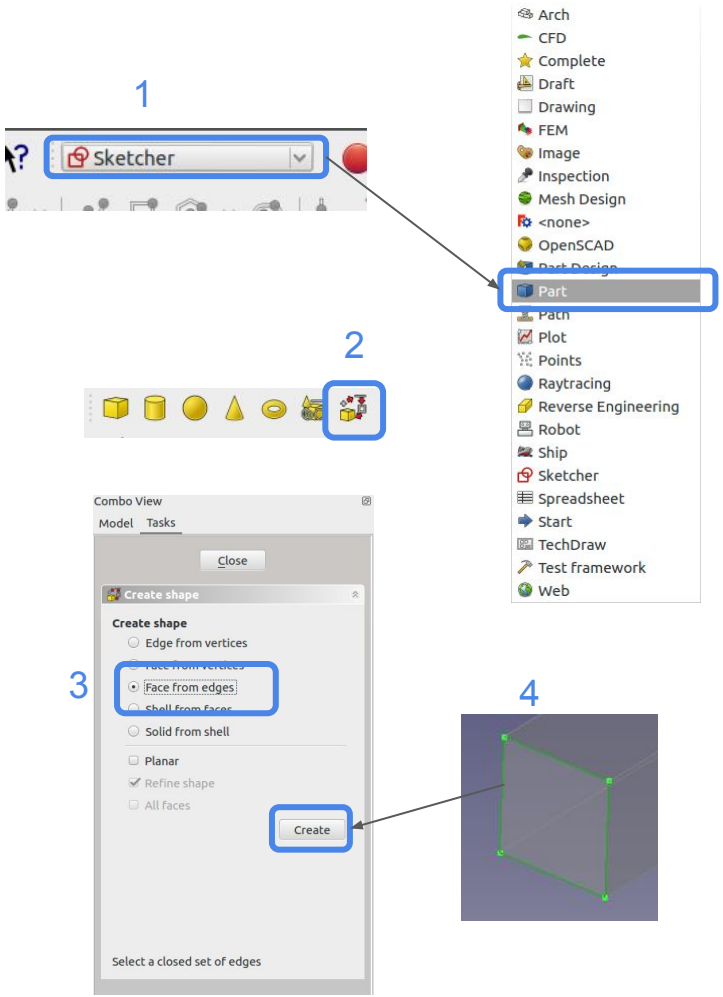
Draw a rectangle

- We start by creating reference lines. These lines link the edges from the already existing Cube object to our newly created sketch.
- To do so, click on the create edge link icon, then select the edges of pre-existing cube face.
- We now draw a rectangle, and make the corners coincident with our reference lines.
- Once all the corners of the rectangle are coincident with the reference lines, there should now be a rectangle of equal size to that of the Cube's face.



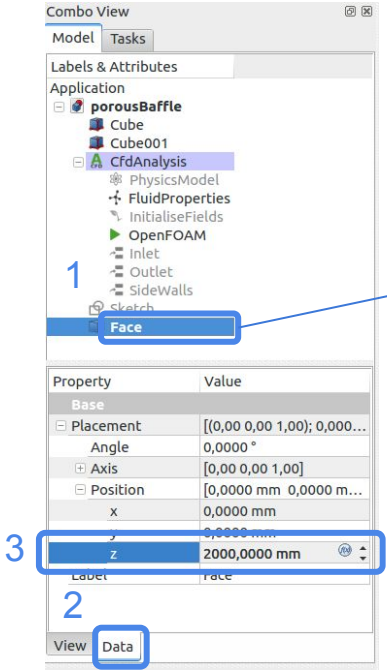
Create face from edges

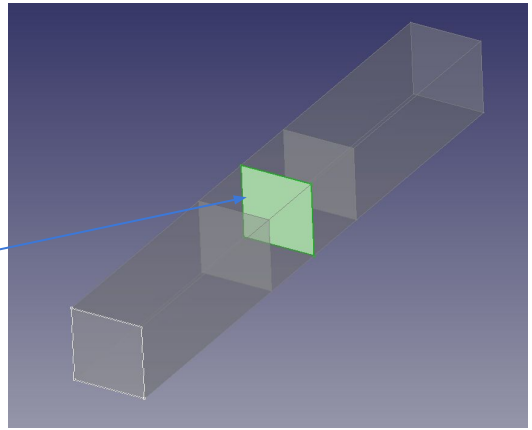
- In order to change our sketch into a face, activate the “Part” workbench.
- Click on the “Advanced shape utility” icon.
- Select “Face from edges” option.
- Select all the edges of our newly created sketch (NOTE: hold in “Ctrl” to select more than one edge simultaneously).
- Click “Create”.



Move face

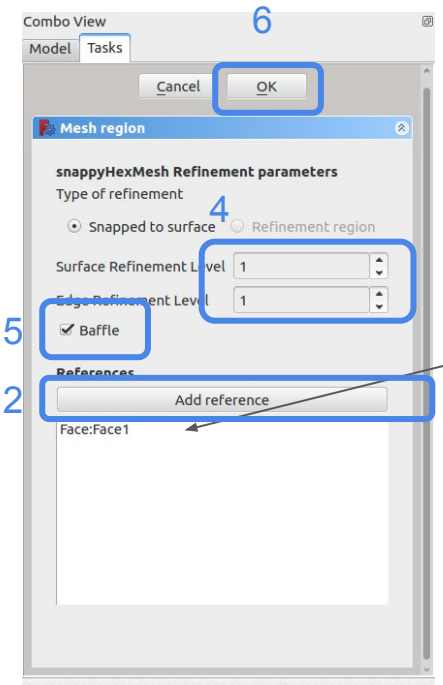
- Move the newly created face to the centre of the 3 cubes. To do so:
  - Highlight the “Face” in the list of objects.
  - Within the “Data” tab, change the z-Position to 2000mm.
- As before, create a snappyHexMesh with a characteristic length of 50mm.

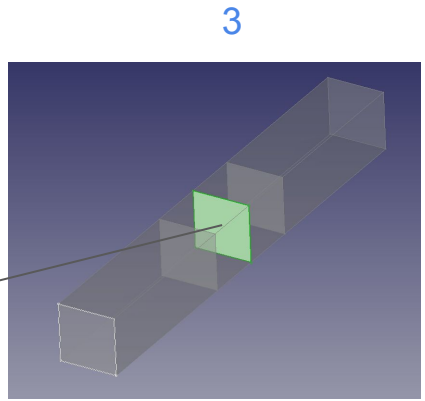




Set mesh baffle mesh region

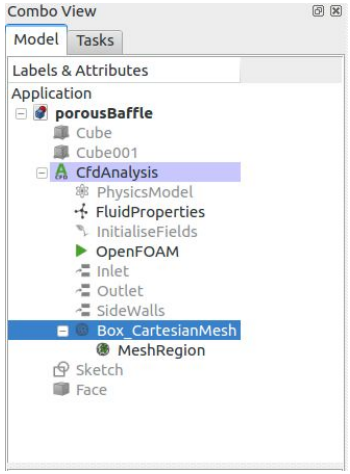
- After having created a snappyHexMesh object, highlight the mesh object and click on the MeshRegion icon.
- As before, click on “Add reference”, select the face and set the levels of refinement.
- Importantly, ensure that the “Baffle” option is selected. By selecting the Baffle option, guarantees that snappyHexMesh generates an internal face of 0 thickness.

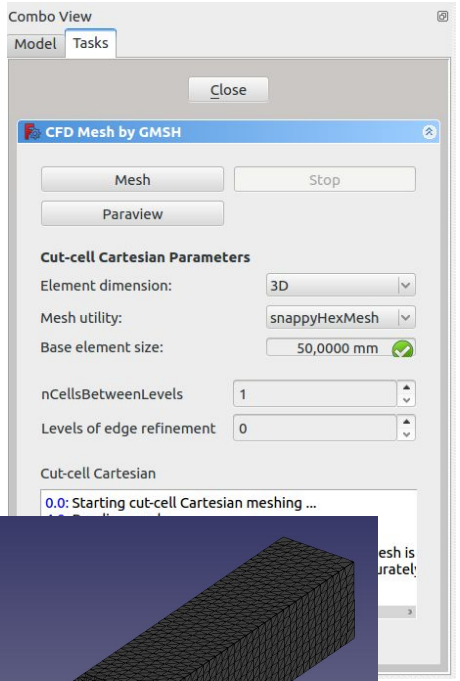


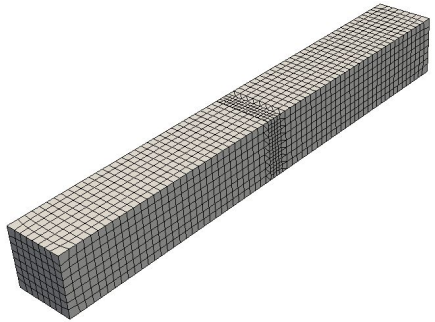


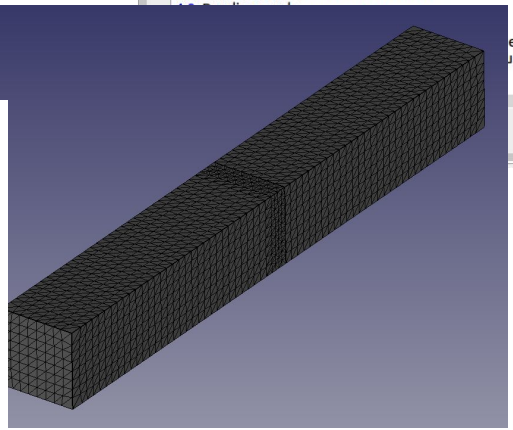
Re-compute the new mesh

- In the list of objects, double-click on the *parent* (“Box_CartesianMesh”) mesh object (or right-click and select “Transform”).
- Click “Mesh” to re-compute the mesh with the new refined region.
- The mesh should now be locally refined in the vicinity of the newly created internal face.







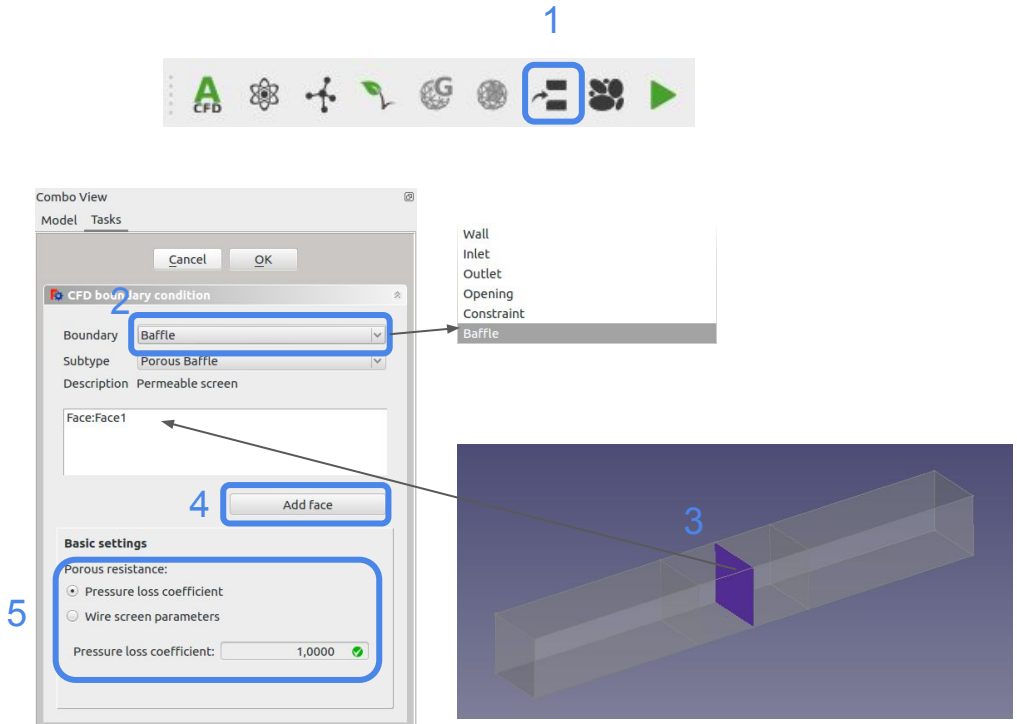


Add porous baffle

- A porous baffle is added by adding the “Baffle” boundary condition.
- Click on the boundary condition icon.
- Select “Baffle” from the Boundary Type.
- Click on the “Face” and click “Add face”
- Specify either the “Pressure loss parameters” or “Wire screen parameters”.

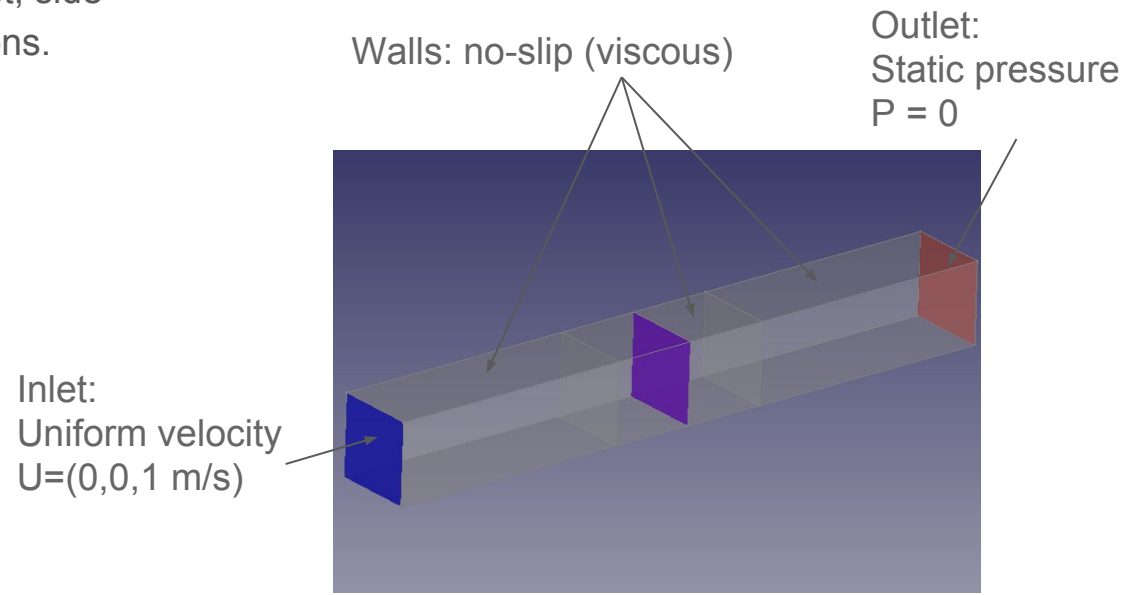
TIP: Recall, that various objects can be made visible or hidden in the “Model” tab by highlighting the objects and pressing spacebar (or right-clicking and selecting Show/Hide). This may be necessary to select the “Face”.

TIP: The baffle Face can alternatively be added by highlighting the “Face” object in the list of objects under the “Model” tab, and then clicking “Add face” within the still open CFD boundary condition “Tasks” tab.



Adding the remaining boundaries

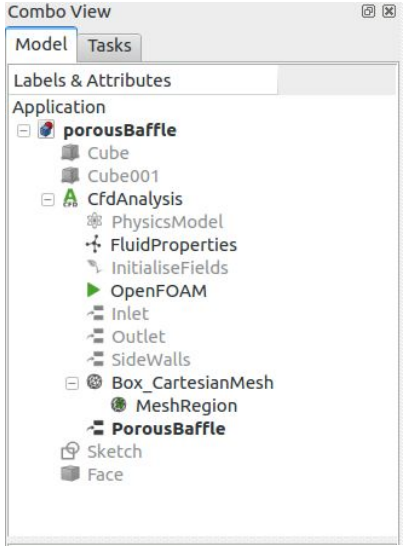
- Following the instructions provided in Tutorial #1, add the remaining inlet, side walls and outlet boundary conditions.
- The remaining settings include:
  - Fluid: air
  - Initialise with potential flow

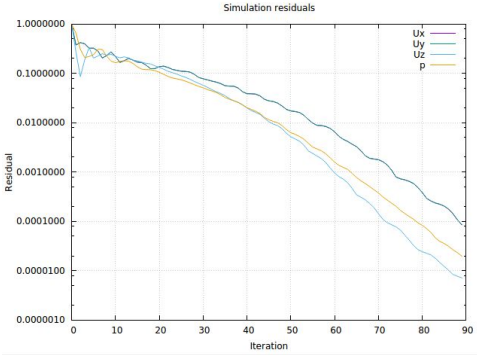


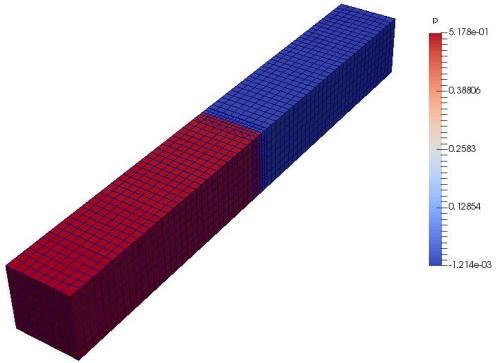
Porous baffle results

- With the problem setup we can now write the test case, and run the solver.
- We show here the residual plot and pressure results for the baffle test problem
- Notice the large distinct pressure drop across the porous baffle.

NOTE: The pressure solved for by OpenFOAM is the density normalised pressure:
p = P/density.







The End